Practical Circuit Board Construction

● Ian 'at-play' with a design under construction.

Ian Liston-Smith has a technique for producing circuit boards for his projects, that look like p.c.b.s, but doesn't use chemicals.

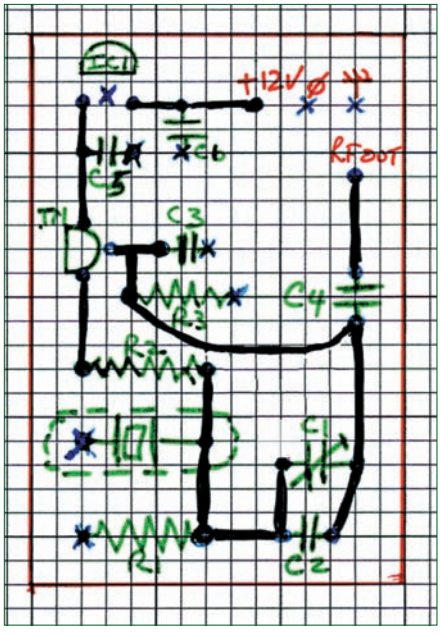
For many decades now the printed circuit board (p.c.b.) has replaced all other assembly methods in the manufacture of electronic goods. But for reasons I have never understood, radio and electronic hobbyists try to emulate this construction method. And why is this you may ask?

The p.c.b. was specifically designed for mass production and I think it is therefore singularly unsuited for the home constructor, unless of course it is provided as part of a kit. When the task of making an individual p.c.b. is well done, the enthusiast can obtain a professional finish.

Making a one-off p.c.b. is a method that is mostly time consuming, inflexible and requires the use of some often unpleasant chemicals. Fortunately various alternatives for creating p.c.b.s, are available in the form of copper strip matrix boards.

Matrix boards with fixed copper strips have two significant disadvantages. Often an awkward component layout is required. The resulting redesign of the layout, for anything other than a simple circuit, can take a considerable time to rework.

Many matrix boards are not generally suitable for circuits operating above medium frequency r.f. or the lowest of the h.f. Amateur bands, due to their lack of earth plane and stray capacitance between the strips or pads.

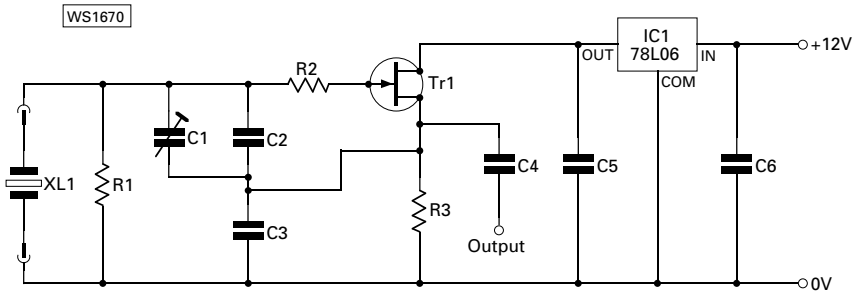


● Fig. 2: If there is no circuit layout in the magazine article, then sketch one out, on squared paper, as shown here.

Dead-bug

The technique of construction, often called the 'dead-bug' (so called because of the look of some of the components) is another favoured construction alternative. This is a method that uses a single sided p.c.b. material board as the earth plane and 0V rail. (*If you're still unsure, have look at many of the smaller projects produced by* ***George Dobbs G3RJV****. It's a method favoured by many other members of the G-QRP club*. Editor)

In many instances, when using the dead-bug method the major components, such as integrated circuits (i.c.s) are glued to the copper clad material, which gave rise to the characteristic look of dead bugs with its legs in the air! The other components are soldered to each other or the copper earth plane.



● Fig. 1: A simple crystal oscillator circuit used to demonstrate the technique presented here.

The dead-bug method, favoured for many of the simpler projects, is ideal for prototyping. It's a method I often use, although I think it tends to look rather too messy for the finished project.

Island Matrix Technique

Another alternative method of construction is the 'island matrix'. This is another interesting method, using very small squares of board, cut out and glued onto a much bigger board of p.c.b. material.

However, the island technique doesn't provide a neat finished layout either. The capacitances formed between each of the islands and the main board also rule this system out for many r.f. projects, except those at the low end of the spectrum.

I was looking for another way - one that combines the ease of production of the other methods, but has a more professional finished look. The method described here is the one I have settled on. I doubt my method is original, but I now use it for all my projects.

In my method, the components are mounted on the upper side of single sided board and the leads pass through countersunk holes and are then wired up underneath. I'm sure that purists would say that this strictly speaking isn't a p.c.b. However, it combines the best of both worlds.

It's a system I have not seen described by anyone else (although years ago I did briefly describe it elsewhere). It's a method that is at least as easy as the alternatives with the following advantages:

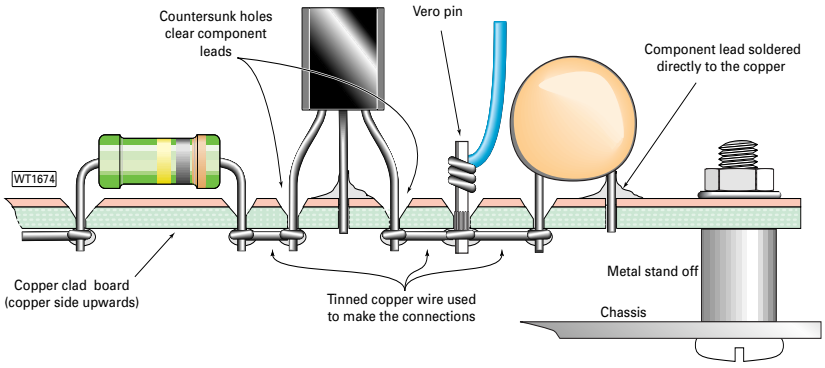

Countersunk holes clear component leads
Vero pin
Component lead soldered directly to the copper
WT1674
Copper clad board (copper side upwards)
Tinned copper wire used to make the connections
Metal stand off
Chassis

Fig. 3: A cross section through a typical board design, just to illustrate the technique of linking components together.

- No etching chemicals are required.
- The upper copper side acts as the earth plane and this is particularly suited to r.f. circuits, although it works equally well for power supplies, audio, digital circuits etc.
- Suitable for the novice or experienced constructor.
- Can easily be modified to accommodate circuit alterations.
- The finished appearance is that of a real p.c.b.
- If a project has a p.c.b. layout, this can be used. The wiring underneath then follows the track design.

Crystal Oscillator

The example design described here shows how a crystal oscillator is constructed using this technique. The circuit is shown in **Fig. 1** and a sketched board layout in **Fig. 2**. But if there is a layout in the magazine article, then it would make sense to follow that layout, rather that redesign it for your board

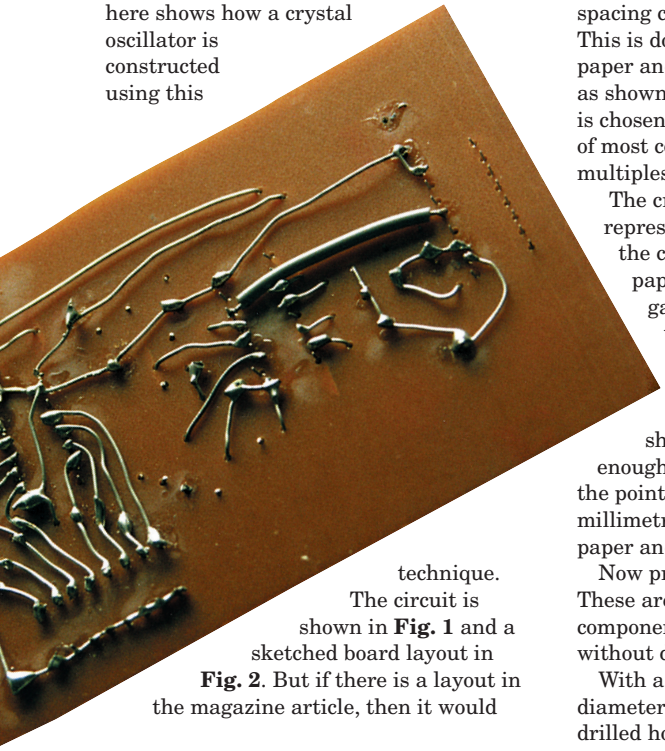


Fig. 6: The project of Fig. 5, but this time from below!

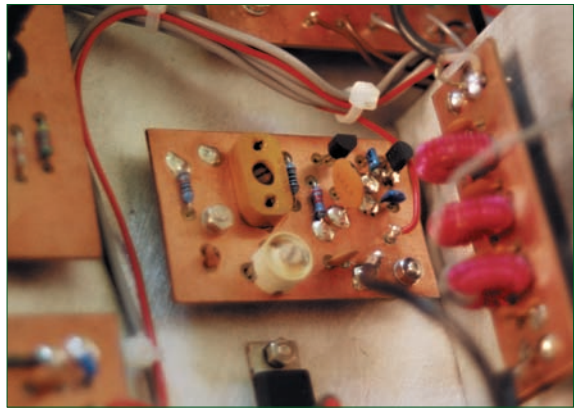


Fig. 4: The finished board in place in the full project.

The Method

Collect all the components for the circuit before starting so, that lead and component spacing can be worked out. This is done on 0.1 inch graph paper and drawn actual size as shown in Fig. 2. This paper is chosen because the spacing of most component leads are multiples of 0.1 inches.

The crosses in the layout represent earth points on the copper side of the board. When the paper layout is completed (allowing gaps between some components for the mounting screws), cut it out and tape it over a piece of board of the same size, copper side up.

With a centre punch or other sharp instrument, press hard enough to make a good indentation at all the points marked with dots. Using a one millimetre drill bit, drill right through the paper and board at these points.

Now press to mark all the X-points. These are the holes for the earthed component leads. Remove the paper without drilling these.

With a Verocutter or six millimetre diameter twist drill, countersink all the drilled holes. These are the holes the component leads must pass through without touching the copper surface.

Now drill through all the X-points. Doing it in this order prevents the X-points from being accidentally countersunk.

There are likely to be minute pieces of copper around the countersunk holes that could short out components later. To clear these, rub the whole board with emery paper or coarse wire wool. This will smooth the holes and polish the copper ready for soldering.

Finally, to remove all the debris that will now have got pushed into the countersunk holes, gently go over them again with the Verocutter or six millimetre drill. The board is now ready for the components.

All components requiring an earthed connection have the appropriate lead soldered to the top as shown in **Fig. 3**. Veropins are used for test points and any flying leads. Tinned copper wire is used to connect the components together under the board. If the layout is cramped, slip an insulated sleeve over any wires running close to each other.

Drill holes for the M3 mounting bolts between the components. These will provide a good earth to the main chassis. The board is held above the chassis with plastic or metal spacers. As you can see from the photographs of finished item in **Fig. 4**, the end result is very neat and professional looking.

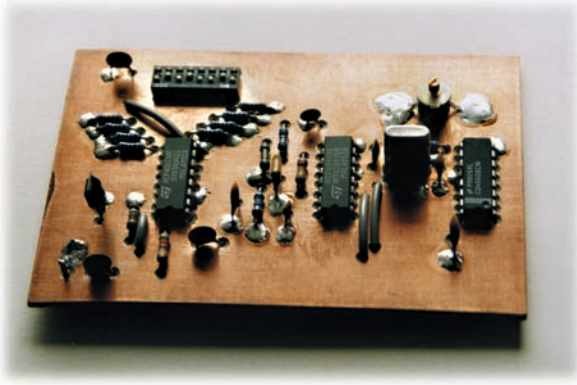


Fig. 5: A rather more complicated layout from above...

More Complicated

It's possible to create rather more complicated circuit layouts that the design I've chosen here to demonstrate the technique. A much more complex circuit using i.c.s and switches (see photographs) illustrate the flexibility of this assembly technique. Have a look at the photographs of **Fig. 5** and **Fig. 6** to see the design. I'm sure you'd agree that from the component side it's difficult to tell that it isn't a well made amateur p.c.b.

No one would know it wasn't a real p.c.b.!

PW